LAMINAR FLOW THROUGH A DOUBLE ELBOW GEOMETRY WITH LATTICE BOLTZMANN NUMERICAL METHOD

N. M. Akmal, C. S. N. Azwadi and M. S. Zuhairi

Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia,
81200 Skudai, Johor
akmaln@uthm.edu.my
azwadi@fkm.utm.my

Abstract

This paper presents a lattice Boltzmann method for the simulation of laminar flow through double elbow geometry. The lattice Boltzmann method was built up on the D2Q9 model and the single relaxation time called the lattice-BGK method. The dependence of reattachment length on the Reynolds number is determined. Results show that the flow was found to be strongly dependent on Reynolds number and two-dimensional behavior for Reynolds numbers below approximately 400. Comparable results were obtained between the present approach and those from a Navier-Stokes solver.

Keywords: Double elbow geometry, laminar fluid flow, lattice Boltzmann method

1. Introduction

The lattice Boltzmann model (LBM) has increasingly been accepted as a viable alternative approach to the well-known finite difference, finite element, and finite volume techniques for solving the Navier-Stokes equations [1, 2, 3]. Unlike any other well-known computational methods, LB scheme treats the fluid behavior at the microscopic level and brings together its information to predict the macroscopic behavior of fluid flow. This provides the opportunity to go deeper into the particles community and understand how the interaction between them would affect the macroscopic parameters of fluid flow. Another important improvement to enhance the computational efficiency is the implementation of the Bhatnagar Gross Krook collision operator (BGK) approximation (single relaxation time approximation) for the collision function [4]. Even under the simplifications provided using BGK, LBM has demonstrated its ability to simulate flows in porous media [1], immiscible fluids [5] and magneto-hydrodynamics [6].

Historically LBM was derived from the lattice gas automata (LGA) method [7]. Consequently, the LBM inherits some features from its precursor, the LGA method. The dynamics of distribution function evolving on a lattice space consists of two main steps; collision, particle at the same site collide according to a set of hard sphere particle collisions rules; and streaming, particle move to the nearest node in the direction of its velocities. However, instead of using Boolean representation of particle in LGA, LBM uses real numbers represent the local ensemble-averaged particle distribution function, and only kinetic equations for the distribution function are solved. The number of discrete velocities determines the lattice structure of LBM models. In other words, the discretization of physical space is coupled with the discretization of momentum space. As a result, computational in LBM is only restricted with uniform lattice structure and second order accuracy in space and time [8].

In this study, we used LBM to investigate the velocity profile and flow characteristics of a fluid flowing through double elbow geometry. This geometry is commonly found in piping systems, involving high accelerations and decelerations when the fluid is flowing at high speeds. The reattachment length after elbows is studied to determine the location of the fully developed profile after flow separation. Pressure gauges, flow meters, anemometers and other auxiliary devices are usually installed outside of the separated region to avoid the flow instabilities that could commonly occur. Pressure loss, low flow rate, leakage, fracture, and loss of cooling are some of the issues that could possibly arise with the use of elbows. Our study of this geometry was based on another, better-known geometry that is the backward-facing step.

Flow over backward-facing step geometry brings together geometric simplicity with a complex flow behavior because of the interaction of a recirculation region and a jet in a confined duct. Their relevance also stems from the fact that they often occur in industry. Recirculation is induced at sudden changes in geometry, in boundary layer flows with severe adverse pressure gradients, and in strong swirling flows.

A detailed experimental study was conducted by Armaly et al. [9] for an expansion ratio close to 2.0 and downstream aspect ratio close to 18. Armaly et al. raised the question of three-dimensionality of step flow. Three-dimensionality manifests itself in a discrepancy in primary recirculation region between experiments and two-dimensional simulations for Reynolds numbers above 400. Armaly et al. suggested that the discrepancy in primary recirculation region could be attributed to the secondary vortex destroying the two-dimensional character of the flow. Armaly et al. also reported that the primary reattachment length increase nonlinearly with Reynolds numbers.

2. Lattice Boltzmann method

In this section, we discuss briefly the theory of the lattice Boltzmann method. Detailed formulation of LBM can be found in the following publication [10].

If a two-dimension nine-velocity model is used (D2Q9) [7] then the time evolution lattice Boltzmann without force can be expressed as

$$f_i\left(\mathbf{x}+\mathbf{e}_i\Delta t, t+\Delta t\right)-f_i\left(\mathbf{x},t\right)=-\frac{1}{\tau}\left[f_i\left(\mathbf{x},t\right)-f_i^{eq}\left(\mathbf{x},t\right)\right] \tag{2.1}$$

In this equation, $f_i(\mathbf{x},t)$ is the single-particle distribution function, $\boldsymbol{c_i}$ is the particle's velocity, and τ is the relaxation time for the collision. The discrete velocity is expressed as

$$\mathbf{c}_i=\begin{cases}\left(0,0\right), & i=0\\ \left(\cos\theta_i,\sin\theta_i,\right), \quad \theta_i=\left(i-1\right)\pi/4, & i=1,3,5,7\\ \sqrt{2}\left(\cos\theta_i,\sin\theta_i,\right), \theta_i=\left(i-1\right)\pi/4, & i=2,4,6,8\end{cases} \tag{2.2}$$

The Boltzmann-Maxwellian equilibrium distribution f^{eq} is expressed as

$$f_i^{eq}=\frac{n}{\left(2\pi RT\right)^{D/2}}\exp\left\{-\frac{\left(\mathbf{c}-\mathbf{v}\right)^2}{2RT}\right\} \tag{2.3}$$

with the weights w_0 = 4/9, $w_1 = w_3 = w_5 = w_7$ = 1/9, and $w_2 = w_4 = w_6 = w_8$ = 1/36. The updating of the lattice consists of two steps: 1) A streaming process, where the particle densities are shifted in discrete time steps through the lattice along the connection lines in direction $\boldsymbol{c_i}$ to their next neighboring nodes and 2) A collision step, where locally a new particle distribution is computed by evaluating the right hand side of equation 2.1. The macroscopic number density n and velocity vector $\boldsymbol{u}$ are related to the distribution function by

$$\begin{aligned} n&=\sum_i f_i\\ n\mathbf{u}&=\sum_i \mathbf{c}_i f_i\end{aligned} \tag{2.4}$$

For single-phase flow, pressure can be calculated from $p=c_s^2 n$ with the speed of sound $c_s=1/\sqrt{3}$.

Through Chapman-Enskog expansion, the above model recovers to the incompressible Navier-Stokes equations

$$\frac{1}{\rho_0 c_s^2}\frac{\partial p}{\partial t}+\nabla\cdot u=0 \tag{2.5}$$

$$\frac{\partial u}{\partial t}+u\cdot\nabla u=-\frac{1}{\rho_0}\nabla p+\upsilon\nabla^2 u \tag{2.6}$$

In the previous equations, ρ_0 is the constant average density in the system and the kinematics viscosity is

$$\upsilon=\frac{2\tau-1}{6} \tag{2.7}$$

In the small Mach number (equivalent to the incompressible limit), the first term in the left hand side in the continuity equation is negligible. Thus, the incompressible Navier-Stokes equations are obtained.

3. Geometry of flow domain and boundary conditions

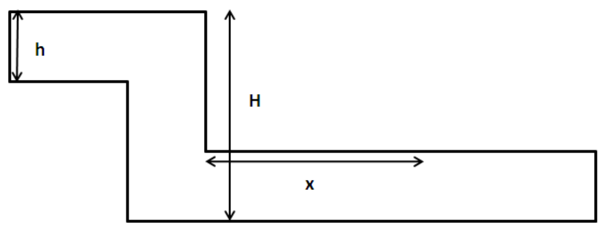

Figure 1 shows a sketch of the geometry of the flow problem used in the present study. The distance between the two elbows relative to their width is termed as the proximity ratio, and is defined by *H/h* where *H* is the total height of the geometry and *h* is the channel height of the inflow channel. The primary recirculation region is expected to occur at the top wall of the channel outlet, as marked by x in Figure 1.

The Reynolds number in this study is defined as

$$\mathrm{Re} = \frac{UD}{\upsilon} \tag{3.1}$$

as in Armaly et al, where U is two-thirds of the maximum inlet velocity, which corresponds in the laminar case to the average inlet velocity, D is the hydraulic diameter of the inlet channel and is equivalent to twice its height, $D = 2h$, and õ is the kinematic viscosity. At the outlet of the computational domain, the flow should be fully developed again. Hence, the application of simple outflow conditions assuming zero gradients of all flow variables is typically sufficient.

For the boundary condition, we applied the so-called 'no-slip bounce back boundary condition' [8] at the upper wall, bottom wall and the vertical step wall. This of boundary condition reflects all distribution functions at boundary sites, back along the link on which they arrived. Averaging the velocity at the boundary, before and after the collision, gives the required boundary conditions at the wall, $u = v = 0$.

4. Results and discussions

We have simulated flow through double elbow geometry and those results are shown in Figure 2. This figure shows streamlines of steady-state flow field for a proximity ratio *H/h* = 3.0. At Reynolds number of 100, recirculation regions of various sizes developed in three of the four corners in the middle of the geometry. However, as mentioned previously, the region of particular interest occurs at the top wall of the outlet channel. With the increase in Reynolds number, the vortex in this region significantly grew in size. Consequently, the reattachment length also increased, until it reaches close to the geometry outlet for the case of Re = 400.

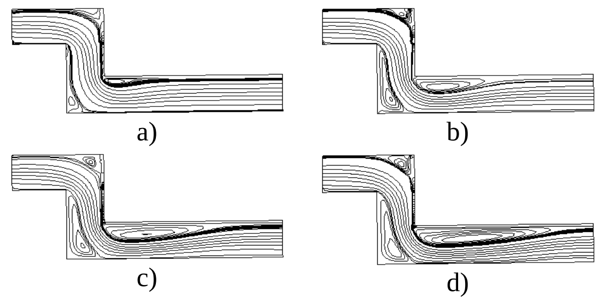

Fig 2: Flow through double elbows (LBM). Proximity ratio *H/h* = 3.0; (a) *Re* = 100; (b) *Re* = 200; (c) *Re* = 300; (d) *Re* = 400

For comparison purposes, the same geometry was constructed using the Gambit 2.2.30 meshing software and simulated using Fluent 6.3.26. All simulation parameters were kept as close to those used in the LBM simulation as possible. The results of these simulations are as shown in Figure 3. The behavior of the fluid flow is somewhat similar in both methods of simulation, but there is a noticeable difference in the length of the recirculation region between the two methods at lower Reynolds numbers.

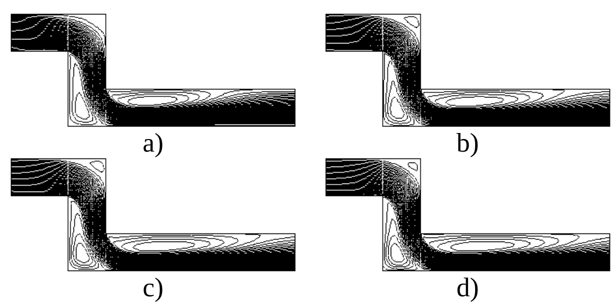

Fig 3: Flow through double elbows (Fluent). Proximity ratio *H/h* = 3.0; (a) *Re* = 100; (b) *Re* = 200; (c) *Re* = 300; (d) *Re* = 400

We have calculated the distance from the flow separation to the reattachment point by calculating the line $U = 0$, and extrapolated along the upper wall of the outlet channel. In Figure 4, we present reattachment length value, which has been normalized by step height, versus Reynolds number for both the LBM and Fluent simulations. In other words, the figure demonstrates the effect of Reynolds number to the reattachment length for the two simulation methods.

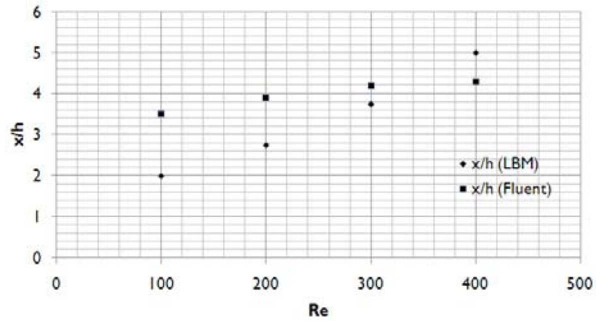

Fig. 4 Reattachment length normalized by step height vs. Reynolds numbers

For Re > 100, the vortex that develops at the top wall of the outlet channel increases along with the increase of Reynolds number, as exhibited by both simulation methods. However, the reattachment length provided by simulations in Fluent at lower Reynolds numbers is significantly larger than that from the LBM simulations. Conversely, the rate of increase for the reattachment length is somewhat smaller in Fluent simulation results compared to those provided by LBM. This discrepancy could be due to the difference in outflow boundary conditions of the two simulation methods, coupled with the effect of possible insufficiency in the length of the outlet channel. Nevertheless, the results of the both simulations agree in terms of the effect of the Reynolds number towards the reattachment length of the recirculation zone.

5. Conclusion

In this study, we used the lattice Boltzmann method to simulate the flow behavior through double elbow geometry. Our study showed that the flow pattern and the length of primary recirculation region are significantly affected by the value of Reynolds number. In our simulations, the reattachment length increases with an increase in Reynolds numbers. The numerical prediction presented in this paper confirmed that the lattice Boltzmann method for flow predictions could be successfully employed to compute flows in this type of geometry, with results comparable to that of a more established Navier-Stokes solver, at least up to Reynolds numbers of approximately 400. However, further work could be done to improve the level of confidence in these simulations and to increase the scope of the study.

6. Acknowledgements

The authors wish to thank Universiti Teknologi Malaysia and Malaysia Government for supporting this research activity.

7. References

[1] Bernsdorf, G. Brenner and F. Durst, “Numerical analysis of the pressure drop in porous media flow with Lattice Boltzmann (BGK) automata”. Comp. Phys. Comm., 129. (2000), pp. 247-255

[2] S. Chen, D. Martinez and W. H. Mattaeus, Phys. Of Plasma, 1 (1994), pp. 1850-1860

[3] C.S. Azwadi and T. Tanahashi, Proc. Asean Cong. Fluid Mech., 11 (2006), pp. 275-280

[4] P. L. Bhatnagar, E. P. Gross and M. Krook, “A model for collision process in gasses 1. Small amplitude process in charged and neutral one component system.” Physics Reviews. 94, (1954), pp. 511-525.

[5] N. S. Martys and H. Chen,” Simulation of multicomponent fluid in complex three dimensional geometries by the lattice Boltzmann method.” Phys. Rev. E 53 (1996), pp. 743-750.

[6] S. Chen, H. Chen, D. Martinez and W.H Matthaeus, “Lattice Boltzmann model for simulation of magnetohydrodynamics.” Phys. Rev Letters 67. (1991), pp. 3776-3779.

[7] U. Frish, B. Hasslacher and Y. Pomeau Phys. Rev. Lett., 56 (1986), pp. 1505-150

[8] C. S. Nor Azwadi and T. Tanahashi,” Simplified Finite Difference Thermal Lattice Boltzmann” Method. Int. J. of Modern Phys.B, Vol. 22, No. 22. (2008), pp. 3865-3876.

[9] Armaly B. F., Durst F., Peireira J. C. F., Schonung B., Experimental and theoretical investigation on backward-facing step flow. *J Fluid Mech*. 1983, 127**,** 473-496.

[10] S. Chen and G. D. Doolen, *Ann. Rev. Fluid Mech.,* (1993) 329